Real-ESSI Simulator

Results Postprocessing and Visualization Manual

José Abell, Sumeet Kumar Sinha, Yuan Feng,

Han Yang, Hexiang Wang

and

Boris Jeremić

University of California, Davis, CA

Version: 13Feb2026, 15:41

http://real-essi.us/

This document is an excerpt from: http://sokocalo.engr.ucdavis.edu/~jeremic/LectureNotes/

please use google-chrome to view this PDF so that hyperlinks work

Contents

Chapter 1

Post Processing for Real-ESSI Simulator

2016-

1.1 Introduction

1.2 Model Results Post-Processing

This chapter describes methodology for post processing simulation results from the Real-ESSI Simulator. Two main approaches are used:

- Plotting time histories of scalar results (described in section 1.3 on page 6), using Python and/or Matlab for:
  - components of displacements, velocities, accelerations, pore fluid pressures, for finite element nodes,
  - components of stress and/or strain at integration (Gauss) points within each finite element,
  - components of section forces for structural finite elements
  - energy input and dissipation for parts or whole of the volume/model, in incremental and/or cumulative form
- Visualization of a part or a complete model for displacements, velocities, accelerations, stress and strain components, sectional forces, energy dissipation through visualization system ParaView, as described in section 1.4 on page 7.

1.3 Time Histories Plotting

Time histories of various scalar results (as listed above) can be extracted from output files, saved in HDF5 (Group, 2020), in a format described in chapter 206, on page 1031 in Jeremić et al. (1989-2025).

An excellent set of Python postprocessing tools was developed by Dr, Konstantinos Kanellopoulos, from ETH Zürich! You can find these tools at his github:

`https://github.com/ConstantinosKanellopoulos/Real-ESSI_postprocessing_tools`.

1.4 Post Processing and Visualization using ParaView

ParaView package http://www.paraview.org/ (Ayachit, 2015) is a very powerful multi-platform data analysis and visualization program available as an Open Source. Paraview can be run on supercomputers to analyze datasets of peta-scale size as well as on laptops for smaller data, and has become an integral tool in many national laboratories, universities and industry, and has won several awards related to high performance computation.

PVESSIReader is a plugin for paraview that integrates Real-ESSI Simulator output to Paraview for visualization. PVESSIReader reads Real-ESSI output file, in HDF5 format, files with extension .feioutput. The plugin works for sequential, parallel as well as remote visualization mode. It has a number of visualization features to visualize stresses, eigen modes, relative displacement, physical groups, energy dissipation, etc.

The installation of both Paraview and PVESSIReader is described in some detail in section 209.8.2 on page 1175 of the main document (Jeremić et al., 1989-2025).

1.4.1 Visualization in ParaView : Features

PVESSIReader has been consistently developed and added with lot of visualization options which are built on ParaView Visualization Toolkit (VTK) framework. This section shows all the visualization options that PVESSIReader offers other than what is available in ParaView. The features are illustrated in subsections below with help of examples in *Examples* folder of PVESSIReader source directory.

Before, looking at the features, it's important to know how PVESSIReader works. PVESSIReader takes Real-ESSI HDF5 output (.feioutput) file format as input and creates a *PVESSIReader* folder inside the HDF5 file. PVESSIReader does this to ensure the visualization to be optimized. The contents inside this folder are not important for any regular user. The contents of "PVESSIReader" folder is shown in Figure 1.1, although it is not important to regular users. User must know that, the plug-ins first creates this folder and then uses the content of this folder for visualization in ParaView. So creation/reading of this folder is the essence to visualization in ParaView.

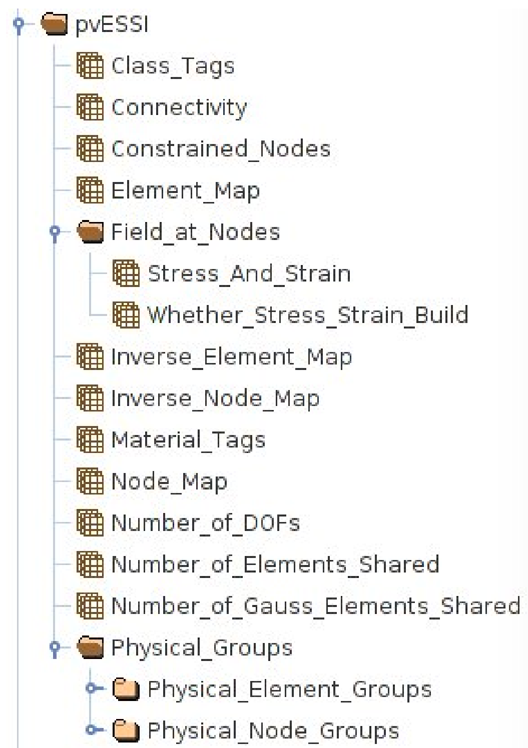


Figure 1.1: Contents of PVESSIReader folder.

PVESSIReader Visualization Options

By default PVESSIReader builds a node mesh. Figure 1.2 shows the various visualization options that is available for PVESSIReader. The Gauss to node interpolation is turned off and other options shown in Figure 1.2 is turned off. As stated in previous Section 1.4.1, the plugin creates the *PVESSIReader folder* inside the output HDF5 file only once and uses it for rest of visualization (even after you close and reopen it). The 'Build PVESSIReader folder' button shown in Figure 1.2 on clicking rebuilds the 'PVESSIReader folder'. If in case the loading of '.feioutput' file fails in ParaView, the user should clicks the 'Build PVESSIReader Folder' before hitting 'Apply' button.

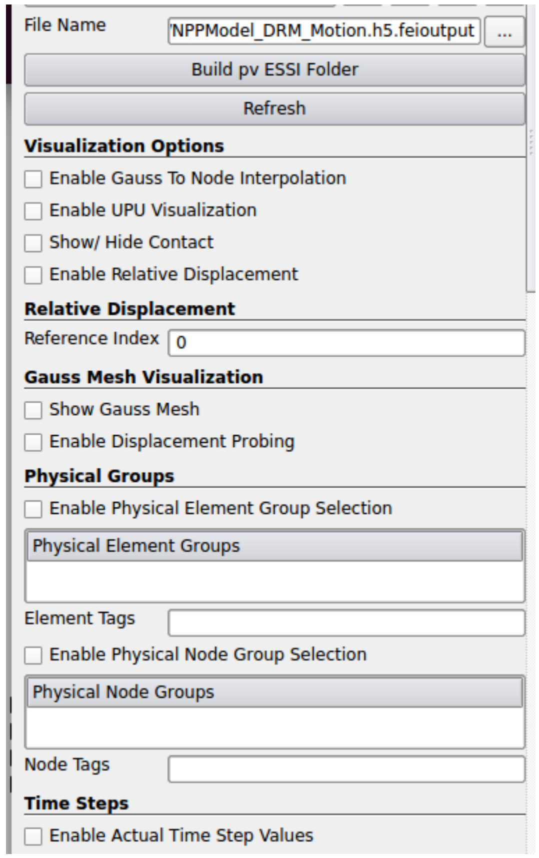
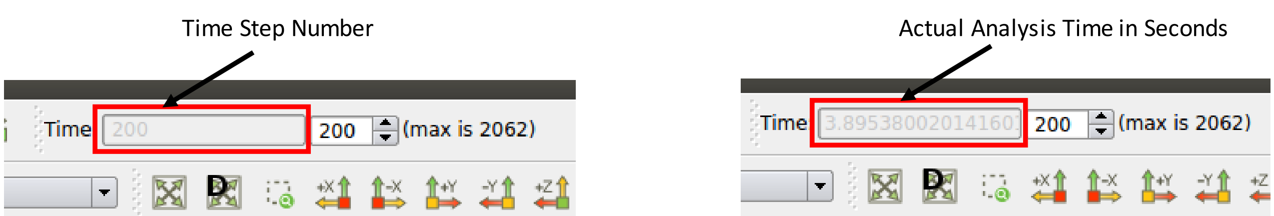


Figure 1.2: PVESSIReader build options.

Description of various PVESSIReader visualization options in the order shown in Figure 1.2 is listed below.

- **Build PVESSIReader Folder** - Rebuild the content of PVESSIReader folder inside output file
- **Refresh** - It reloads the current visualization view.
- **Enable Gauss to Node Interpolation** - Enables the interpolation from Gauss points to node using shape functions. It works only for 8 node and 27 node brick with 8 and 27 Gauss points respectively. See Section 1.4.1.
- **Enable uPU Visualization** - Enables the visualization of fluid displacements (U) and pore-pressure (P) at nodes. See Section 1.4.1.

- **Enable Relative Displacement** - When this is enabled, the displacement of any time step can be visualized with respect to any other reference time step. See Section 1.4.1.
- **Reference Time Step No** -This option is used to set the reference time step number about which the relative displacements would be visualized. See Section 1.4.1.
- **Show Gauss Mesh** - This option can be enabled to visualize the Gauss points (mesh) of the entire model. See Section 1.4.1.
- **Enable Displacement Probing** - This option only works if Gauss mesh option is enabled. With this option, displacements are calculated at Gauss points using ParaView interpolation functions for each elements containing those Gauss points. See Section 1.4.1.
- **Physical Groups** - This option is enabled to visualize pre-defined physical groups in Real-ESSI input or manually defined selected nodes or elements. See Section 1.4.1.
- **Enable Actual Time Step Values** - By default instead of actual simulation time (in seconds), time step number of analysis is provided to ParaView VCR. Figure 1.3 shows the result of enable/disable of this option.



(a) Actual Time Step Values Disabled

(b) Actual Time Step Values Enabled

Figure 1.3: Illustration of difference between enable and disable of `actual time step values`.

In the Figure 1.3, the enabled option gives the exact simulation time of $3.895s$. Whereas, disabling the option shows time step number of 200.

The visualization gets automatically updated on enable/disable of options. When one hit's apply, the corresponding changed result gets updated i.e. the mesh would get real time updated with enable/disable of these options.

Sequential Visualization

Sequential visualization means visualizing the Real-ESSI output on a single core of laptop/desktop. This is used for single output files that Real-ESSI produces for sequential runs. Figure 1.4 shows the visualization of output file produced by sequential Real-ESSI simulation.

```
cd pvESSI/Examples
ParaView ShearBox_Sequential.h5.feioutput
```

The parallel output files of ESSI can also be visualized sequentially. Each individual (core) file can be sequentially visualized showing only a part of the model results. Also all the parallel files at once can be opened as well in ParaView as shown in Figure 1.5. All PVESSIReader examples can be obtained at http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples.

1. To open one core output in sequential

```
cd pvESSI/Examples
ParaView ShearBox_Parallel.h5.1.feioutput
```

2. To open all cores output in parallel

```
cd pvESSI/Examples
ParaView ShearBox_Parallel.h5.feioutput
```

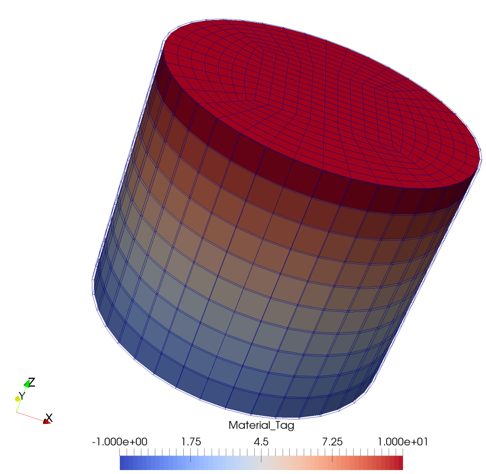
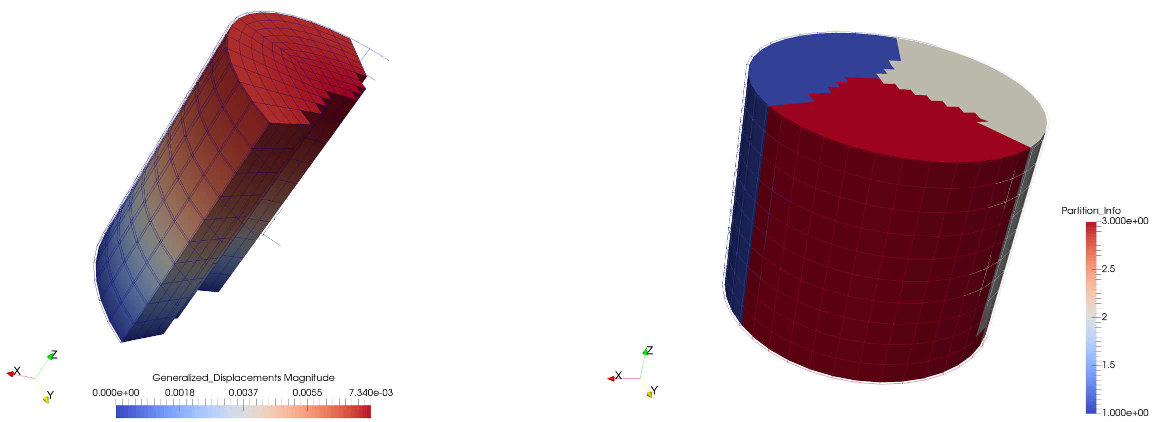


Figure 1.4: Sequential Visualization of output produced by sequential Real-ESSI simulation.



(a) Visualizing only one slave output file

(b) Visualizing all slave output files at once

Figure 1.5: Sequential Visualization of output produced by parallel Real-ESSI simulation.

Remote Visualization

Remote visualization is an important feature that ParaView offers. This is an important feature in need, when simulations are run on super computers with thousands of cores. The steps for remote visualization are shown below with an example on local desktop.

1. Run pvserver on server

```
$pvserver
Waiting for client...
%Connection URL: cs://sumeet:11111
%Accepting connection(s): sumeet:11111
Connection URL: cs://jeremic:11111
Accepting connection(s): jeremic:11111
```

2. Open ParaView on client side and click on **connect** button located on top left window.

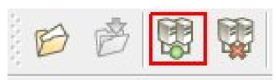
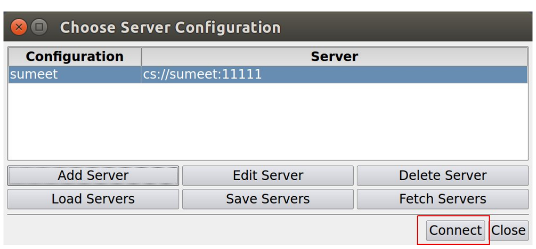
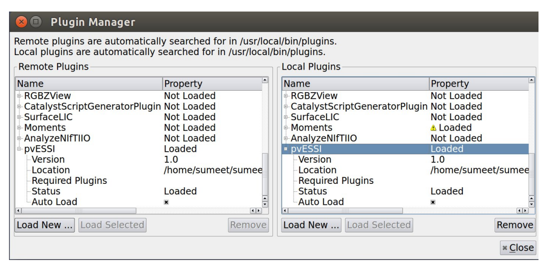


Figure 1.6: Connect Server.

3. Select and connect to the server and then load the plugins on both client and server side as shown in Figure 1.7



(a) Connect to Server



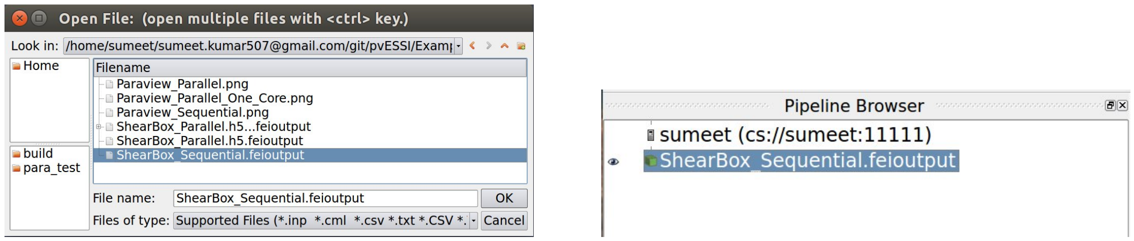
(b) All Core Outputs

Figure 1.7: Connect to the server and load plugins.

4. Navigate to *pvESSI/Examples/ShearBox_Parallel.h5.1.feioutput* and hit apply

Parallel Visualization

Parallel visualization is similar to remote visualization. The only difference is to start the **pvserver** in parallel on multiple cores. It is recommended to have the same number of cores that was used in Real-ESSI parallel simulation. Below shows steps on how to do parallel visualization in ParaView.



(a) Locating file on server side

(b) Client Side Pipeline

Figure 1.8: ParaView Remote Visualization.

```
$ mpirun -np $(nop) pvserver
# $(nop) is replaced by number of cores on which parallel ↩
   visualization is to be run.
```

The next following steps are same to that of Remote visualization as shown in Section 1.4.1. Thus, parallel visualization can be performed remotely as well as locally.

General Field Visualization

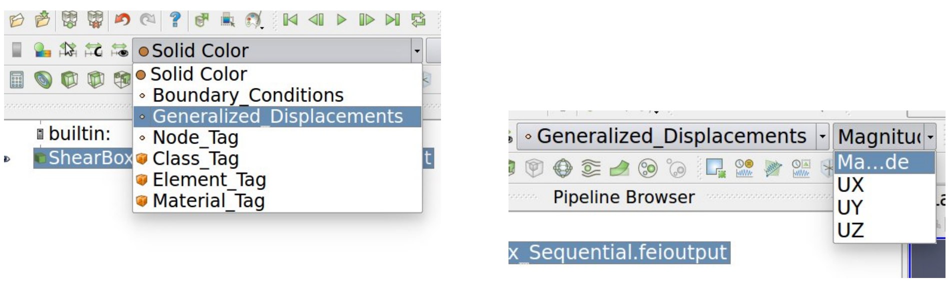
Below is the list of general visualization variables available for any model in ParaView using PVESSIReader plugin. The following subsections describes each option through an example. The examples can be found in pvESSI/Examples directory. The example file '`ShearBox_Sequential.h5.feioutput`' can be downloaded here.

```
cd pvESSI/Examples
ParaView ShearBox_Sequential.h5.feioutput
```

Displacement Field : The displacement field represents the total displacement from the start of the Real-ESSI simulation. There are two modes of displacement field visualization available in ParaView.

NOTE: Please remember to change step number from 0 to any other step number, as all output, including displacements, is 0.0 at step 0.

1. **Scalar field visualization :** The options available are each individual displacement vector components u_x, u_y and u_z in x,y,z directions respectively. It also shows the displacement magnitude $|u| = \sqrt{u_x^2 + u_y^2 + u_z^2}$. The units of displacements field in $[m]$.

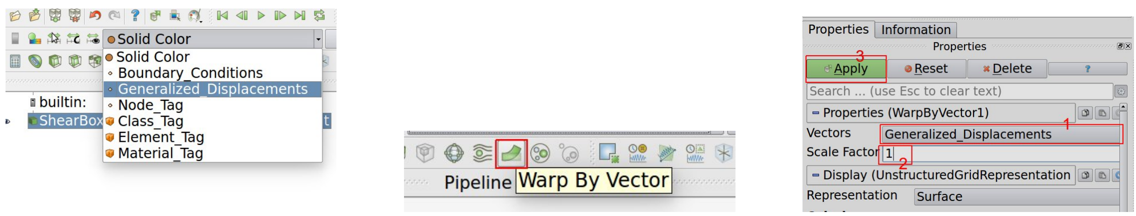


(a) Select Displacement Field

(b) Select scalar field

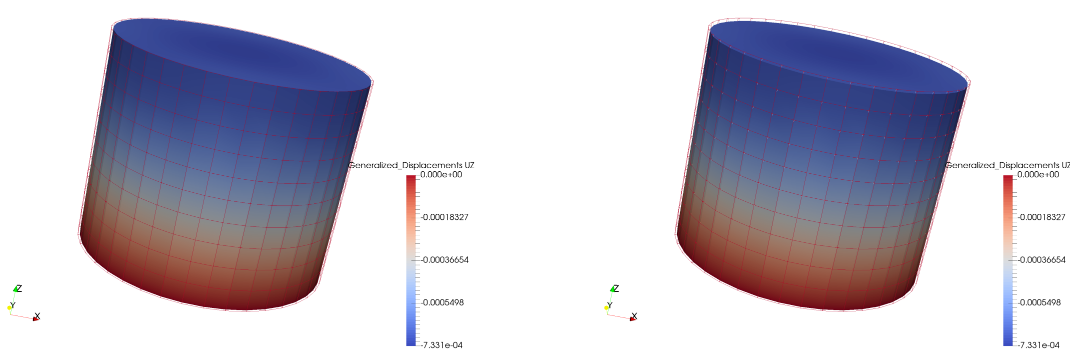
Figure 1.9: Displacement Scalar Field Visualization.

2. **Vector Field visualization :** This is achieved using '`Wrap by Vector`' plugin available in ParaView. Figure 1.10 and Figure 1.11 shows steps to visualize deformed mesh.



(a) Select Displacement Field (b) Select plugin (c) Plugin properties

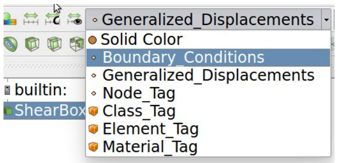
Figure 1.10: Deformation Visualization.



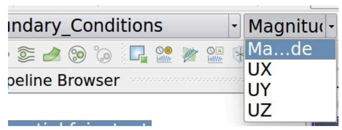
(a) Displacement Field Visualization in Uz (b) Deformation Visualization

Figure 1.11: Displacement field visualization in ParaView.

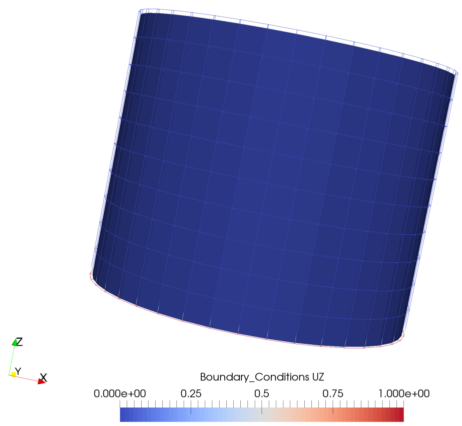
Boundary Conditions: Again this a vector field which contains information about boundary conditions i.e fixities applied in u_x, u_y and u_z directions. A value of **1** means the node is fixed while **0** means it is free. Figure 1.12 shows steps to visualize boundary conditions.



(a) Select Boundary Conditions



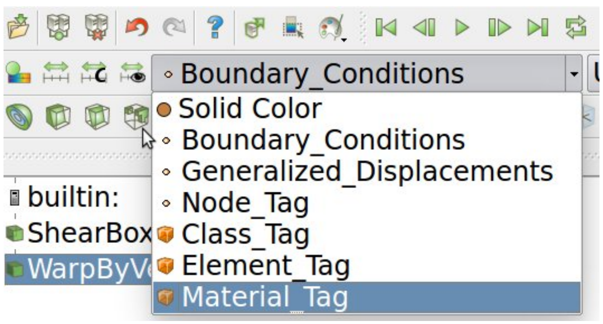
(b) Select Fixities Type



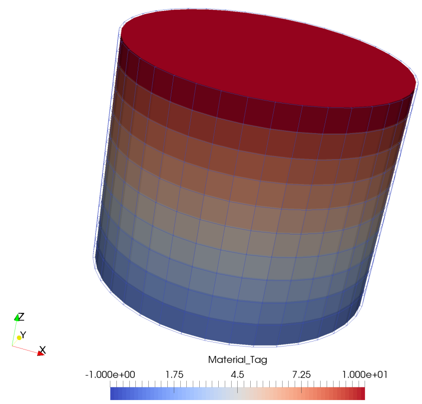
(c) Boundary Conditions in Uz

Figure 1.12: Boundary Conditions Visualization.

Material Tag Visualization : This is a scalar field visualization that shows the *material tag no* associated with the elements in Real-ESSI simulation. Figure 1.13 shows steps to visualize element's material tag.



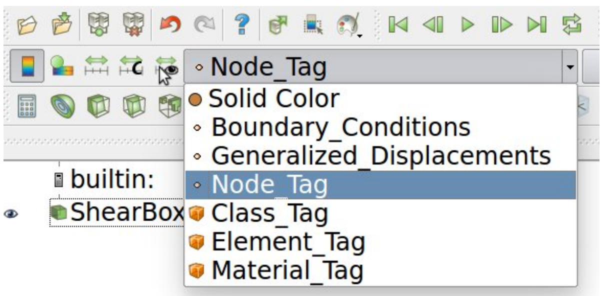
(a) Select Material Tag



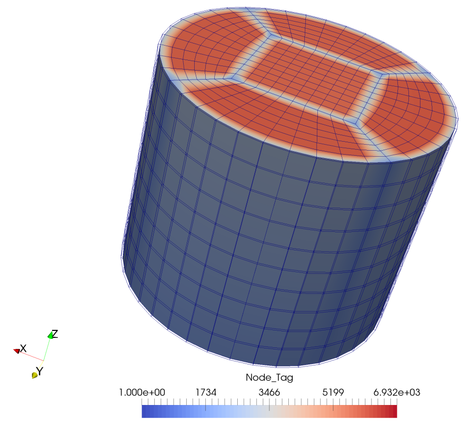
(b) Material Tag Field

Figure 1.13: Material Tag Visualization.

Node Tag Visualization : This is a scalar field visualization that shows the `node no` associated with the nodes in Real-ESSI simulation. Figure 1.14 shows steps to visualize node's tag.



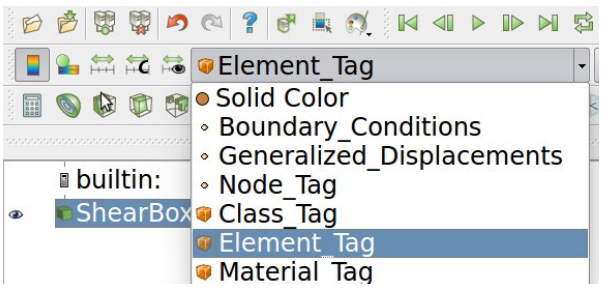
(a) Select Node Tag



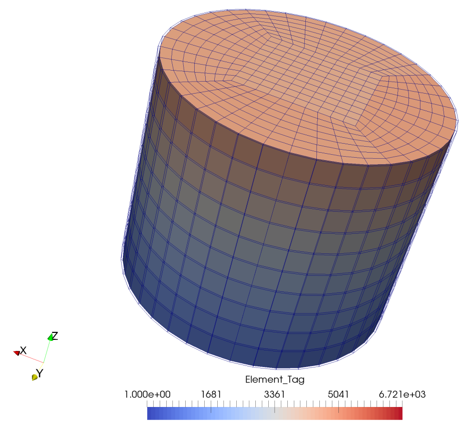
(b) Node Tag Field

Figure 1.14: Node Tag Visualization.

Element Tag Visualization : This is a scalar field visualization that shows the `element no` associated with the elements in Real-ESSI simulation. Figure 1.15 shows steps to visualize element's tag.



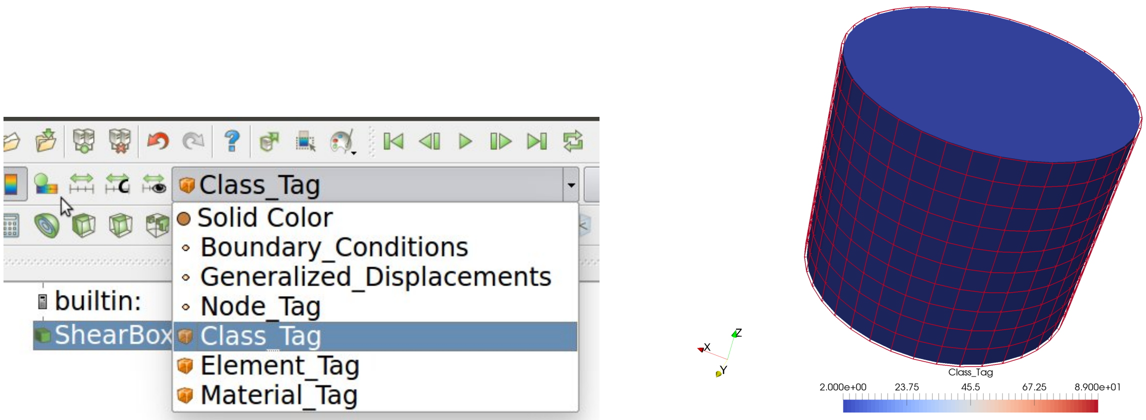
(a) Select Element Tag



(b) Element Tag Field

Figure 1.15: Element Tag Visualization.

Element Class Tag Visualization : This is a scalar field visualization that shows the **Element's Class Tag** number associated with the each *element type* in Real-ESSI simulation. Figure 1.15 shows steps to visualize element's tag. Section 1.5.4 shows the class tag for various element types available in Real-ESSI.



(a) Select Class Tag (b) Class Tag Field

Figure 1.16: Class Tag Visualization.

Relative Displacement Visualization

When the ‘`Enable Relative Displacement`’ is checked, the relative displacement visualization option becomes active. BY default, the relative displacement time step number is set to ‘0’ as shown in Figure 1.17. Time step number ‘0’ corresponds to initial conditions of the loading stage output file.

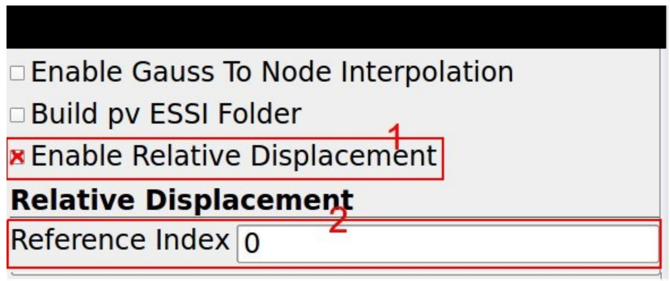


Figure 1.17: Enable Relative Displacement.

Reference Time Step Number - It defines the relative time step index number for relative displacement visualization. By default it is set to 0 i.e. to the initial conditions. Its very useful, when one wants to visualize deformation coming from the stage itself. For example:- Separating self-weight from Static Pushover Analysis in the Shear Box simulation. The steps to do the same is shown below. The example file `ShearBox_PushOver.h5.feioutput` can be downloaded here.

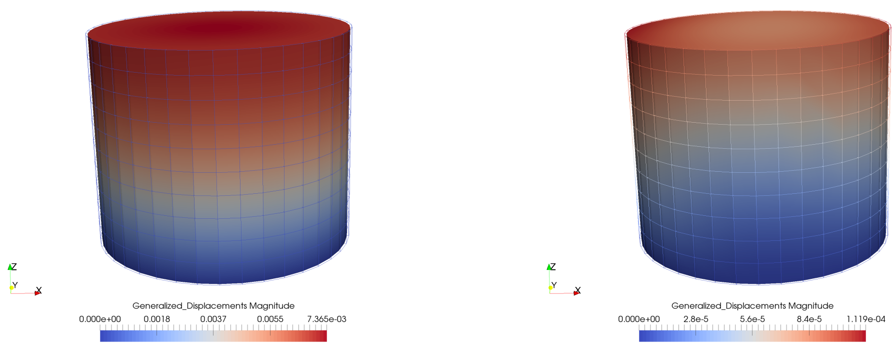
1. Open an example in ParaView

```
cd pvESSI/Examples
ParaView ShearBox_PushOver.h5.feioutput
```

2. Check on `Enable Relative Displacement` under visualization options

3. Apply `warp by vector` plugin and follow the steps as shown in Figure 1.10

Figure 1.18 shows the visualization with and without relative displacement.



(a) Without Relative Displacement

(b) With Relative Displacement

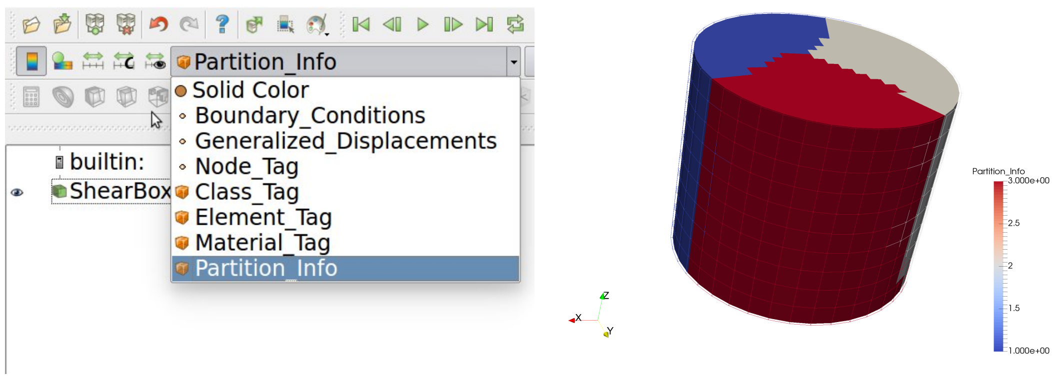
Figure 1.18: Pushover analysis of Shear Box after self-weight load application.

Visualizing Element's Partition

If the ESSI simulation was run in parallel mode, it becomes important to visualize the elements distribution between different cores. In ParaView, one can see the element distribution by selecting "`Partition Info`". Following is shown in Figure 1.19 an example to visualize mesh partitioning. All the example files can be obtained here.

```
cd pvESSI/Examples
ParaView ShearBox_Parallel.h5.feioutput
```

and then select **Partition_Info** as shown below in Figure 1.19



(a) Select Partition Info (b) Mesh Partitioning

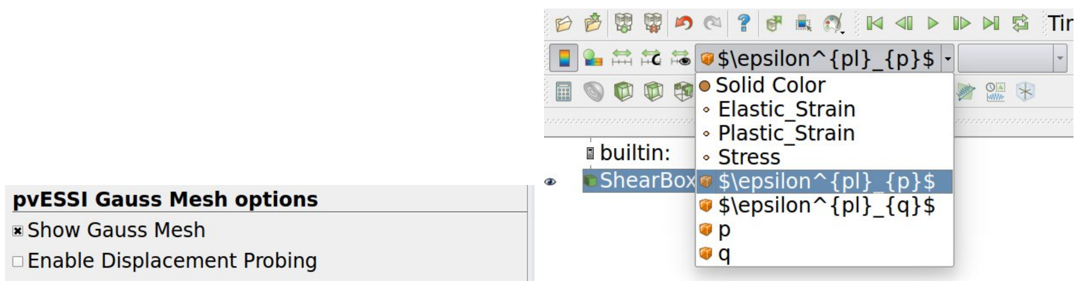
Figure 1.19: Visualizing mesh partitioning.

Gauss Mesh Visualization Options

Often, it is required to visualize stress and strain fields. Since stress or strains are evaluated at Gauss points in 3-D elements, Gauss mesh is needed to visualize them. PVESSIReader offers option to visualize Gauss mesh and it's fields.

- **Show Gauss Mesh** - Shows only Gauss mesh with Gauss attributes.
- **Enable Displacement Probing** - When this option is enabled, displacements are probed to the Gauss location. Its useful in the situation, when one wants to visualize the change in stress with deformation. With this as active, one can apply '`warp by vector`' filter.

It must be noted that the `Enable Displacement Probing` options only works when `Show Gauss Mesh` mode is enabled. Figure 1.20 shows the steps to visualize Gauss mesh.



(a) Gauss Mesh Options (b) Gauss Mesh Fields

Figure 1.20: Visualizing Gauss mesh and it's fields.

The various fields that can be visualized in Gauss mesh mode as shown in Figure 1.20 are shown below.

- **Total Strain** ϵ **:** It defines the total strain from the start of the simulation. It has six independent component $\epsilon_{xx}, \epsilon_{xy}, \epsilon_{xz}, \epsilon_{yy}, \epsilon_{yz}$ and ϵ_{zz}. The magnitude of the total stress in ParaView is defined as $\sqrt{\epsilon_{ij} : \epsilon_{ij}}$.
- **Total Plastic Strain** ϵ^{pl} **:** It defines the total plastic strain from the start of the simulation. It has six independent component $\epsilon^{pl}_{xx}, \epsilon^{pl}_{xy}, \epsilon^{pl}_{xz}, \epsilon^{pl}_{yy}, \epsilon^{pl}_{yz}$ and ϵ^{pl}_{zz}. The magnitude of the total plastic strain in ParaView is defined as $\sqrt{\epsilon^{pl}_{ij} : \epsilon^{pl}_{ij}}$.
- **Total Effective Stress** $\sigma\prime$ **:** It defines the total effective stress from the start of the simulation. It has six independent component $\sigma'_{xx}, \sigma'_{xy}, \sigma'_{xz}, \sigma'_{yy}, \sigma'_{yz}$ and σ'_{zz}. The magnitude of the total effective stress in ParaView is defined as $\sqrt{\sigma'_{ij} : \sigma'_{ij}}$. The unit of visualization is in $[Pa]$.

- **Total Mean Effective Stress** p **:** It defines the total mean of the effective stress σ' from the start of the simulation. It is defined as $p = -\sigma'_{ii}/3$ as described in Equation **??**. The unit of Visualization is in $[Pa]$.

- **Total Deviatoric Effective Stress** q **:** It defines the deviatoric invariant of the total effective stress σ' from the start of the simulation. It is defined as $q = \sqrt{3J_2}$ as described in Equation **??**. Where, J_2 is the second invariant of the deviatoric stress tensor $s_{ij} = \sigma'_{ij} - \sigma'_{kk}/3\delta_{ij}$. The unit of visualization is in $[Pa]$.

- **Total Mean Plastic Strain** ϵ_p^{pl} **:** It defines the mean total plastic strain ϵ^{pl} invariant from the start of the simulation. It is defined as $\epsilon_p^{pl} = -\epsilon_{ii}^{pl}/3$. This visualization parameter is unit-less.

- **Total Deviatoric Plastic Strain** ϵ_p^{pl} **:** It defines the deviatoric invariant of the total plastic strain ϵ^{pl} from the start of the simulation. It is defined as $\epsilon_p^{pl} = \sqrt{3J'_2}$. Where, J'_2 is the second invariant of the deviatoric plastic strain tensor $e_{ij}^{pl} = \epsilon_{ij}^{pl} - \epsilon_{kk}^{pl}/3\delta_{ij}$. This visualization parameter is unit-less.

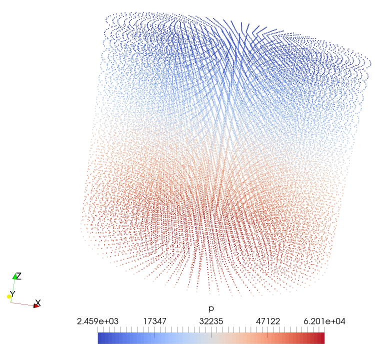
1. Open an example in ParaView. All the example files can be obtained at `http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples`.

```
cd pvESSI/Examples
ParaView ShearBox_PushOver.h5.feioutput
```

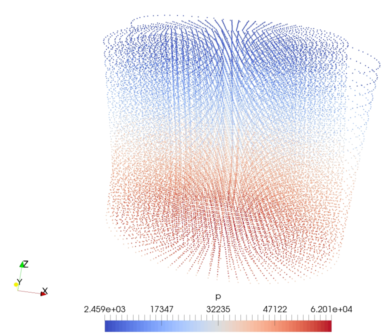
2. Check on *Enable Relative Displacement* under PVESSIReader build options

3. Enable Gauss mesh as shown in Figure 1.20(a). Select `Mean Effective Stress` p `[Pa]`. The resulting visualization is shown in Figure 1.21(a).

4. Enable `displacement probing` as shown in Figure 1.20(a). Apply a `warp by vector` filter and select the vector `displacement` as shown in Figure 1.10. Now select again the `Mean Effective Stress` p `[Pa]` field option to visualize. The resulting visualization is shown in Figure 1.21(b).



(a) Gauss Mesh



(b) Gauss Mesh with displacement probing

Figure 1.21: Visualization of mean effective stress p invariant in Gauss mesh.

Gauss To Node Interpolation Mode Visualization

This visualization mode can be enabled by checking the '`Gauss To Node Interpolation`' option as shown in Figure 1.22(a). In this mode, the total effective stress σ'_{ij}, total strain ϵ_{ij}, total plastic strain ϵ^{pl}_{ij}, total mean effective stress p, total deviatoric effective stress q, total mean plastic strain ϵ^{pl}_{p} and total deviatoric plastic strain ϵ^{pl}_{q} are interpolated from the Gauss points to the nodes of individual element. Individual shape functions of the element (with full Gauss integration) are used to obtain the stress or strain field at nodes. To smooth out the jumps in stress or strain field at the node by adjacent elements, unweighted averaging is performed. For the elements (usually structural) with no Gauss points, the stress or strain contribution at nodes are considered as zero. While taking the averaging, their contributions are not taken, as Real-ESSI does not output stress/strain for them.

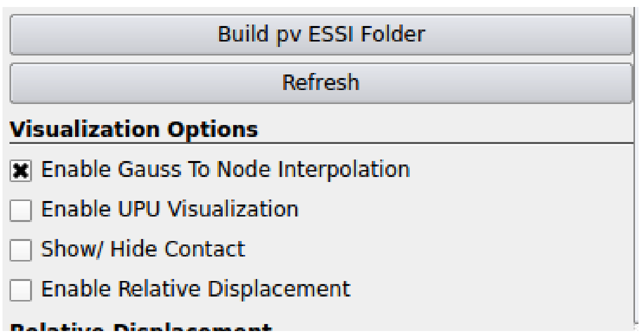
In this mode, visualization of all the parameters listed and described in Section 1.4.1 is available. Figure 1.22 show the steps to enable and use `Gauss to Node Interpolation` option.

1. Open an example in ParaView. All the example files can be obtained at http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples.

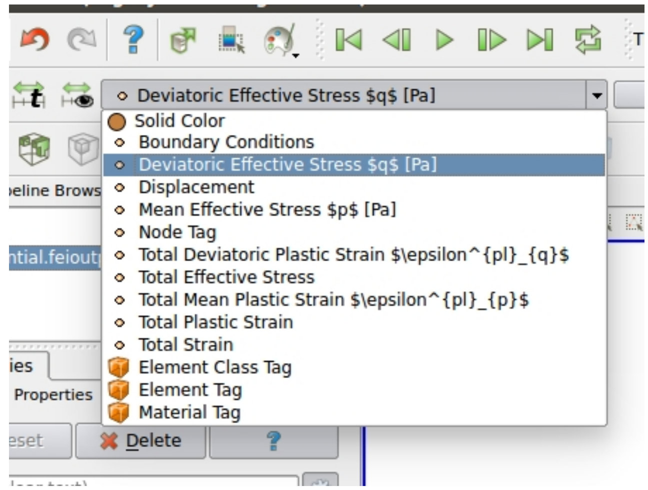
```
cd pvESSI/Examples
ParaView ShearBox_Sequential.feioutput
```

2. Follow the steps as shown in Figure 1.22

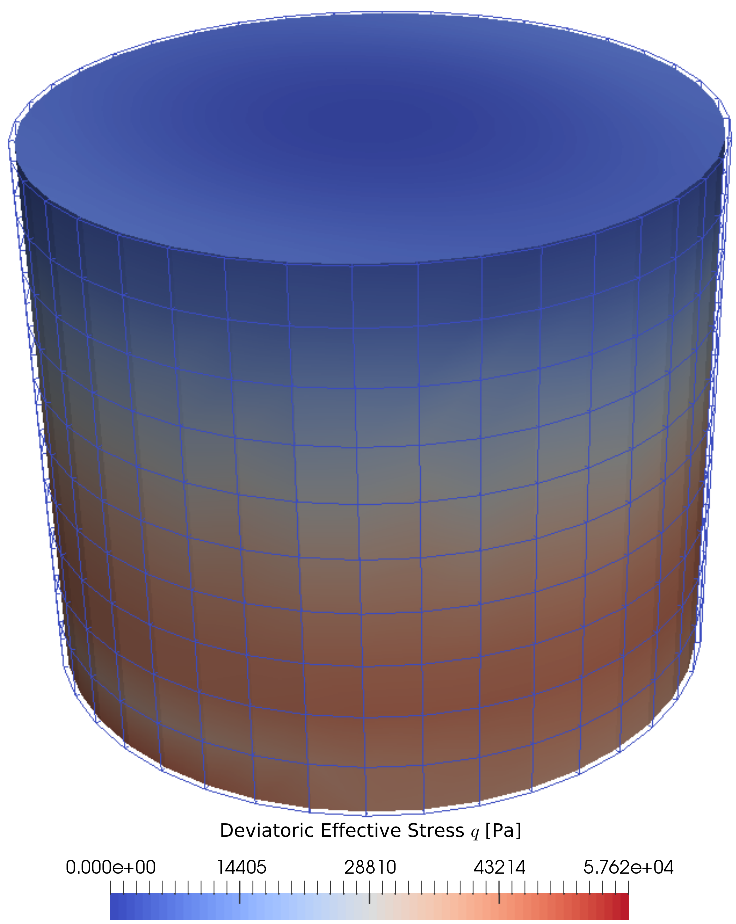
Note : The option `Gauss to node interpolation` is provides only an approximate estimate for stress and strains at nodes. The values obtained at nodes is not accurate and thus `Gauss Mesh Visualization` option described in Section 1.4.1 must be performed to get the accurate stress and strains at Gauss points. Also, it must be noted that this option works only for 8 node brick with 8 Gauss points and 27 node brick with 27 node points. For elements which have less number of nodes that Gauss points, the total number of equations (unknowns) is not equal to constraints (knowns). In this case, only the shape function defined at the nodes are used to get the stress or strain back to the node.



(a) Enable Gauss To Node Visualization Option



(b) Select the visualization variable



(c) Visualization of Deviatoric Stress interpolated to nodes

Figure 1.22: Steps to visualize stress and strain interpolated from Gauss points to nodes.

upU Visualization

This mode is to visualize the upU elements used in Real-ESSI simulation. Enabling this mode, produces additional outputs of 'Pore Pressure $p[Pa]$' and 'Fluid Displacement $U_x[m]$, $U_y[m]$ and $U_z[m]$' at nodes. These additional outputs are described below.

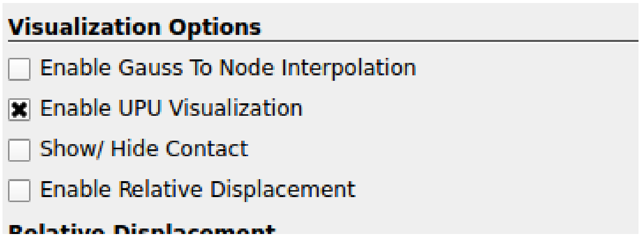
- **Pore Pressure** $p[Pa]$ **:** It defines the pore-fluid pressure in the upU element at the nodes. The magnitude of the pore pressure is $[Pa]$.
- **Fluid Displacement** $U[m]$ **:** It defines the displacement by the fluid particles of upU at nodes. The units is in meters $[m]$. The solid displacement is termed as u and refers to the 'Displacement u' variable in visualization as described in Section 1.4.1.

Since general dry elements does not have any fluid, enabling this option would produce 'zero' pore fluid pressure and fluid displacements at nodes. Below is shown an example that shows how to use the upU visualization feature. Figure 1.23 shows the steps.

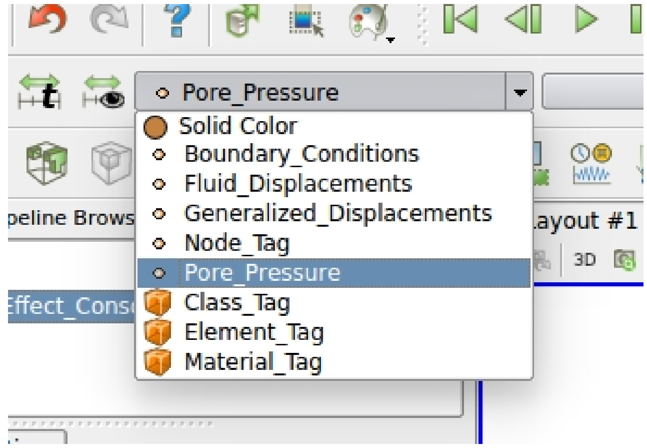
1. Open an example in ParaView. All the example files can be obtained at http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples.

```
cd pvESSI/Examples
ParaView upU_Visualization_Example.feioutput
```

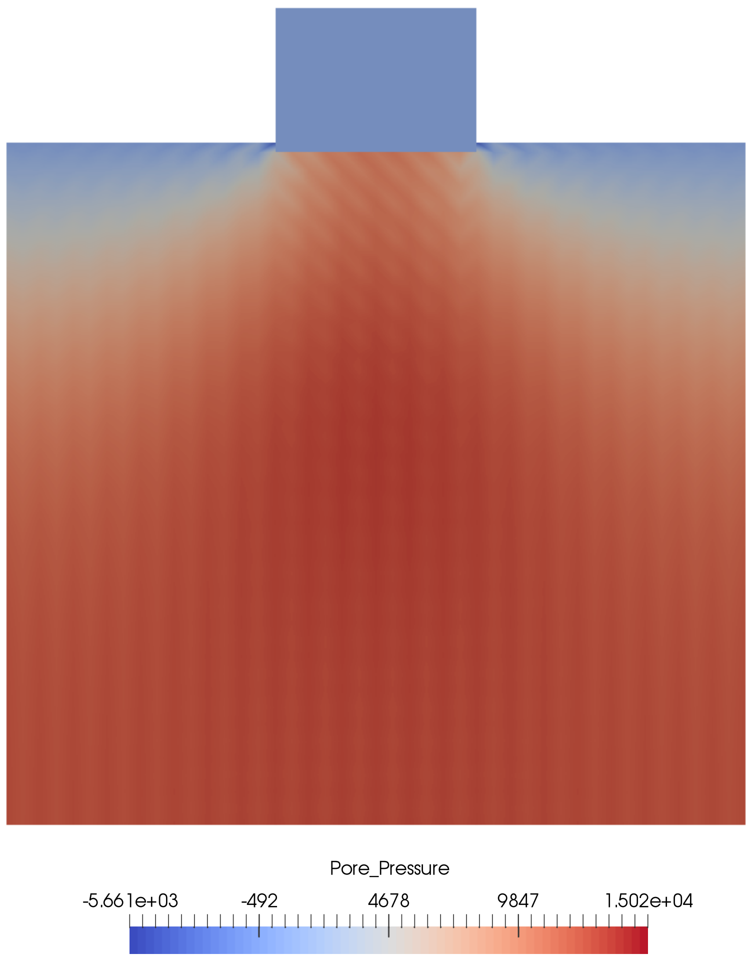
2. Follow the steps as shown in Figure 1.23



(a) Enable upU Mode Visualization Option



(b) Select the visualization variable either pore pressure or fluid displacement



(c) Visualization of Pore Pressure p at nodes

Figure 1.23: Steps to visualize pore pressure p or fluid displacements U in upU visualization mode.

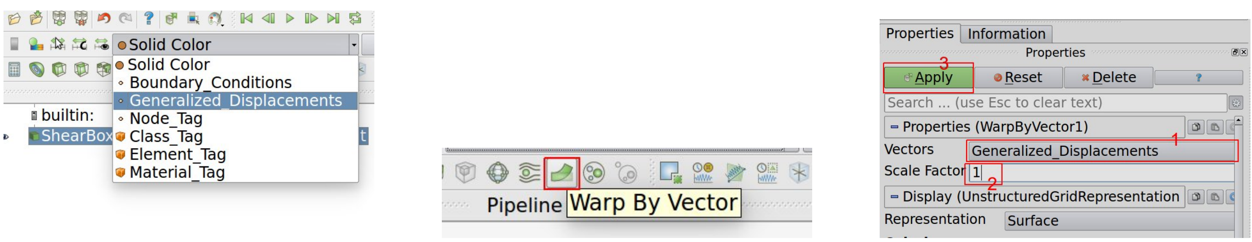
Eigen Mode Visualization

Visualization of eigen modes is that same as visualizing "**displacements**" and applying "**warp by vector**" filter on *Eigen Value Analysis* output of Real-ESSI simulation.

1. Open an *eigen value analysis* output. All the example files can be obtained at http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples.

```
cd pvESSI/Examples
ParaView ShearBoxWall_Eigen_Analysis.h5.feioutput
```

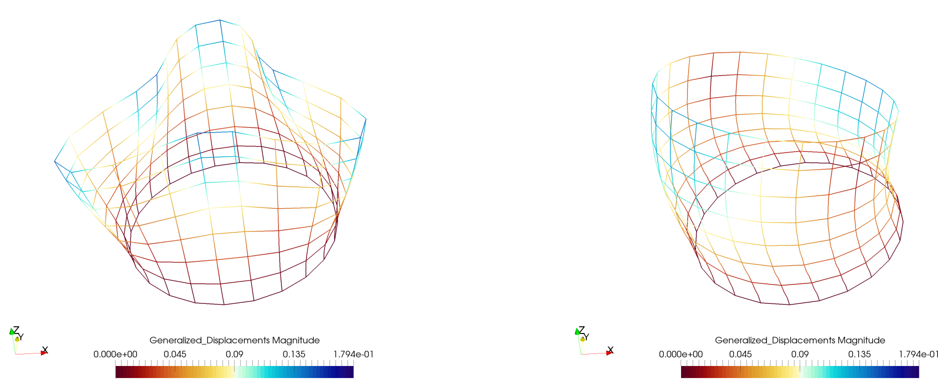
2. Select **displacement** field and then apply **warp by vector** plugin and selected its properties



(a) Select Displacement Field (b) Select plugin (c) Plugin properties

Figure 1.24: Eigen Modes visualization.

3. Now n^{th} time steps here, corresponds to the n^{th} eigen mode.



(a) Eigen Mode 5 (b) Eigen Mode 9

Figure 1.25: Few eigen modes.

Visualizing Physical Node and Element groups

In Real-ESSI it is possible to define different physical groups, for nodes and for elements. If one has defined physical groups in Real-ESSI, you can visualize the same in ParaView. There are two sections here that shows all the physical groups (nodes and elements) defined in the model as shown in Figure 1.26. Section 1.3.4 and Section 1.3.4 shows how to define and add physical group of nodes and elements respectively in Real-ESSI. All the example files can be obtained at http://sokocalo.engr.ucdavis.edu/~jeremic/lecture_notes_online_material/Real-ESSI_pvESSI/Examples.

```
cd pvESSI/Examples
ParaView Model_With_Physical_Groups.h5.feioutput
```

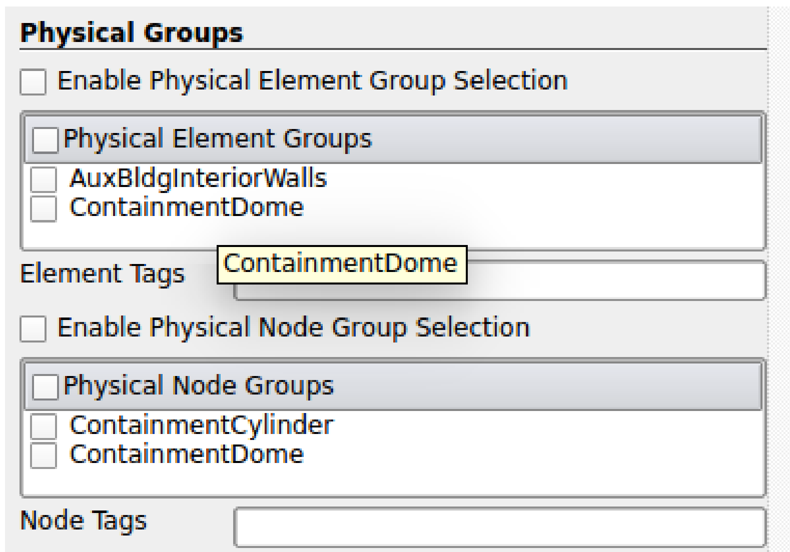
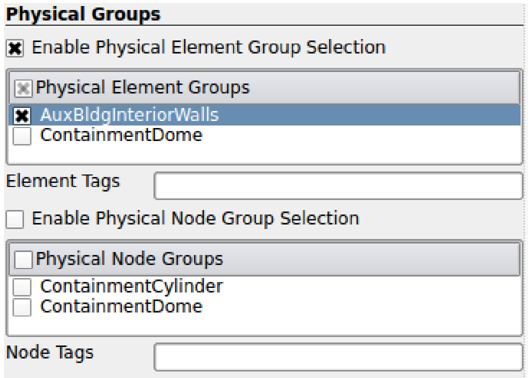


Figure 1.26: Physical Group Visualization Options.

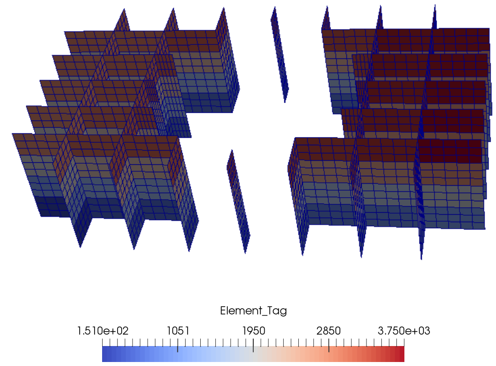
This feature is very useful, when one is interested only in some specific regions of the model than the whole model. Also, This feature becomes very useful, for complicated "interested region/parts of the mesh", which cannot be selected by usual clip/box/..etc filters

1. **Enable Physical Element Group Selection** - Enables the selection of Physical Element Group. By default, it is disabled and one would see the whole mesh. By enabling it, one would only see the selected `'Physical Element Groups'`.

2. **Physical Element Groups** - It shows all the physical element groups defined in Input file of Real-ESSI. The user can select (one or more) of physical groups and hit apply to visualize them. It would show any effect only if the above options **Enable Physical Element Group Selection** is checked. Figure 1.27 shows steps to visualize the physical element groups defined in input files using Real-ESSI DSL.



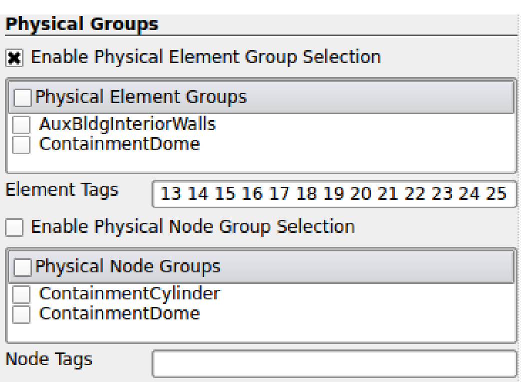
(a) Select `'Physical Element Group Selection'` Option



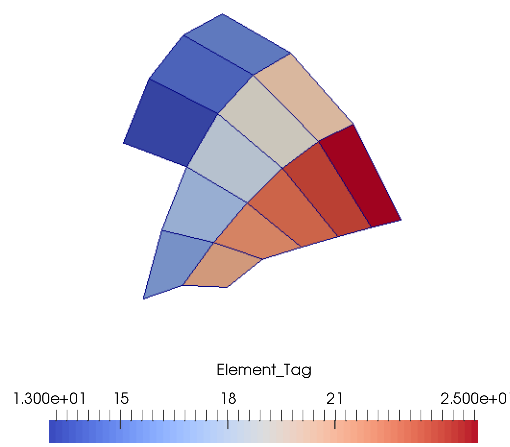
(b) ElementGroup : Auxiliary Building Interior Walls

Figure 1.27: Visualization of physical groups predefined in input file using Real-ESSI DSL.

3. **Element Tags** - This options provides user and interface to manually write element tags to be visualized. The user should enter the element tags against this option as a integer list separated by space. For example:- `'2 10 12 13 16'`, where each of the number corresponds to the element tag defined in the model. Again, this option would only work if **Enable Physical Element Group Selection** option is checked. Figure 1.28 shows steps to visualize elements defined manually.



(a) Select `'Physical Element Group Selection'` Option



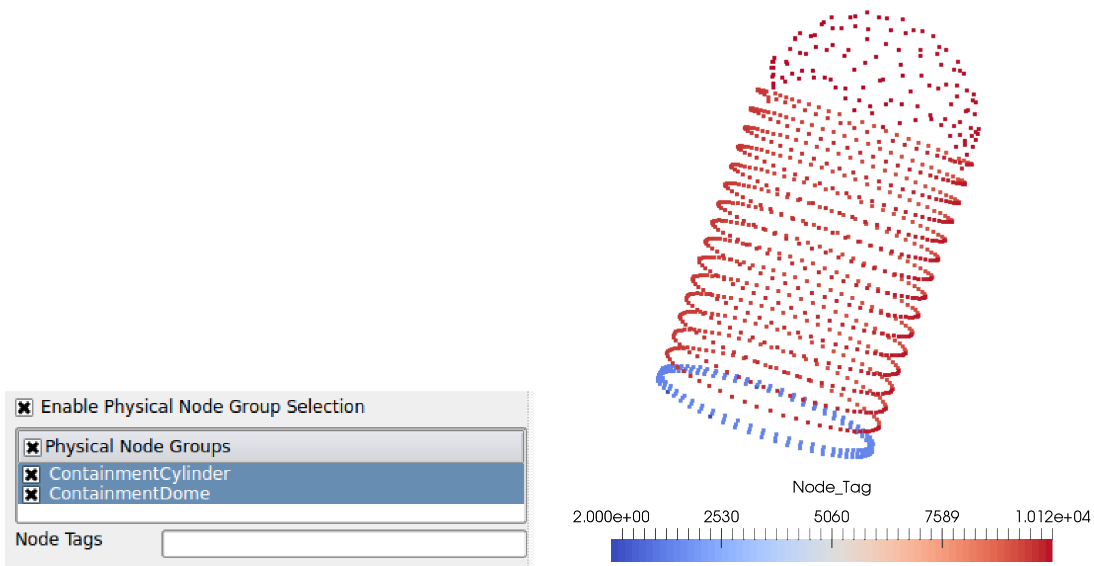
(b) ElementGroup : Manually Defined Element Tags

Figure 1.28: Visualization of physical element group manually defined using PVESSIReader option.

4. **Enable Physical Node Group Selection** - Enables the selection of Physical Node Group. By default,

it is disabled and you would see the whole mesh. By enabling it you would only see the selected Physical Node Groups.

5. **Physical Node Groups** - It shows all the physical node groups defined in Input file of Real-ESSI. The user can select (one or more) of physical groups and hit apply to visualize them.It would show any effect only if the above options **Enable Physical Node Group Selection** is checked. Figure 1.29 shows steps to visualize the physical element groups defined in input files using Real-ESSI DSL.



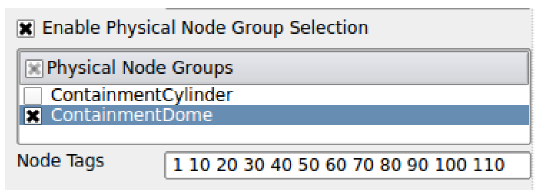
(a) Select `'Physical Node Group Selection'` Option

(b) NodeGroup : Auxiliary Building Interior Walls

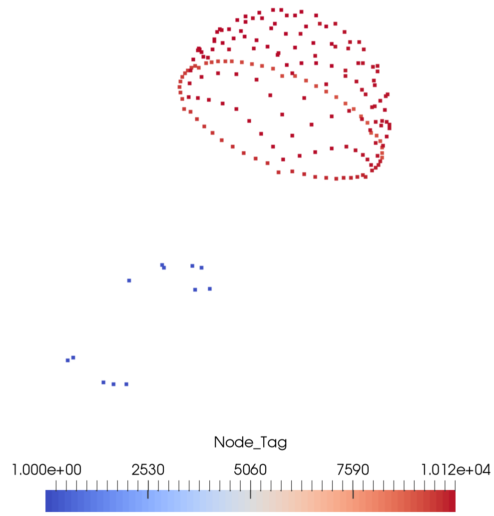
Figure 1.29: Visualization of physical groups predefined in input file using Real-ESSI DSL.

6. **Node Tags** - This options provides user and interface to manually write node tags to be visualized. The user should enter the node tags against this option as a integer list separated by space. For example:- '2 10 12 13 16', where each of the number corresponds to the node tag defined in the model. Again, this option would only work if **Enable Physical Node Group Selection** option is checked. Figure 1.30 shows steps to visualize nodes defined manually.

NOTE: The user can also select both at once, i.e physical element group and physical node group, from the above menu. Figure 1.31 shows mixed selection.

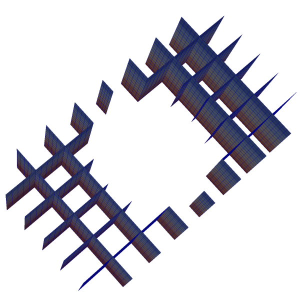


(a) Select `‘Physical Node Group Selection’` Option

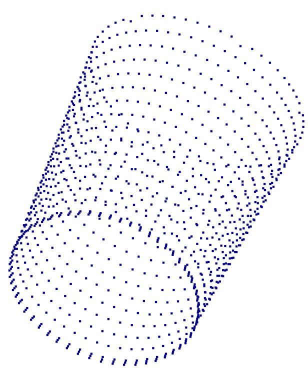


(b) NodeGroup : Manually Defined Node Tags

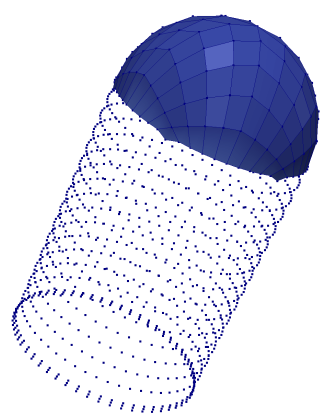
Figure 1.30: Visualization of physical node group manually defined using PVESSIReader option.



(a) ElementGroup : Auxiliary Building Interior Walls



(b) Node Group : Containment Cylinder

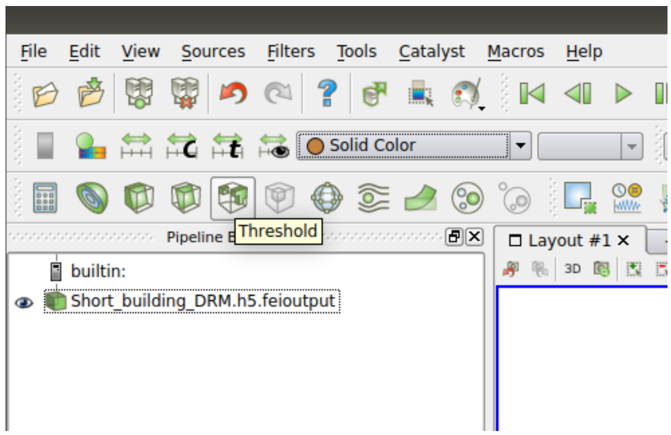


(c) Mixed Group : Containment Cylinder with Dome

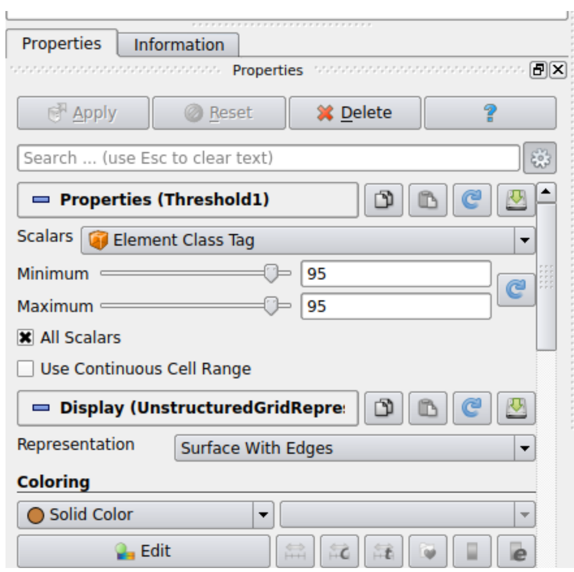
Figure 1.31: Visualization of physical groups.

Using Threshold to Visualize Certain Elements

ParaView allows user to choose specific element types and only visualize selected elements. This function is achieved using **Threshold**. As shown in Figure 1.32, first click on the **Threshold** button in toolbar. Then, choose **Element Class Tag** in the drop-down list of **Scalars**, which can be found in **Properties**. A certain range of **Element Class Tag** can be chosen by setting the minimum and maximum values. If the minimum and maximum values are the same, only one element type will be selected and visualized.



(a) Click on Threshold



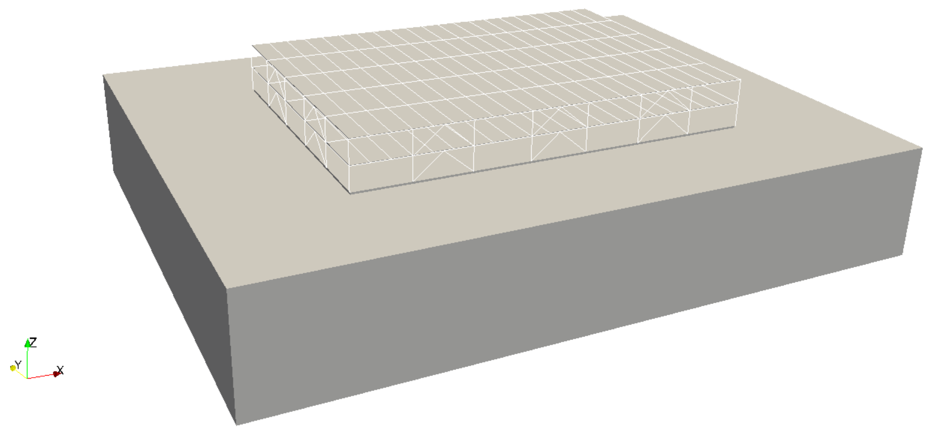
(b) Choose Element Class Tag

Figure 1.32: Using Threshold to Visualize Certain Elements.

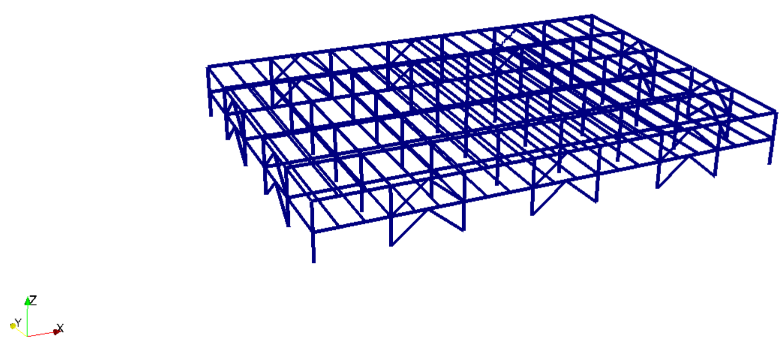
A list of available element class tags in Real-ESSI is provided in Table 1.1.

Table 1.1: Available element class tags in Real-ESSI.

Finite Element Name	Element Class Tag
Truss Element	88
Shear Beam Element	93
Elastic Beam–Column Element	89
Timoshenko Elastic Beam–Column Element	129
Elastic Beam-Column Element with Lumped Mass	90
3D Displacement Based Fiber Beam-Column Element	95
4 Node ANDES Shell with Drilling DOFs	92
3 Node ANDES Shell with Drilling DOFs	91
Super Element Linear Elastic Import	9904
8 Node Brick Element (Order One, Two, Three, Four, Five, Six)	2 (14, 26, 38, 50, 62, 74)
8 Node Brick u-p Element (Order One, Two, Three, Four, Five, Six)	3 (15, 27, 39, 51, 63, 75)
8 Node Brick u-p-U Element (Order One, Two, Three, Four, Five, Six)	4 (16, 28, 40, 52, 64, 76)
20 Node Brick Element (Order One, Two, Three, Four, Five, Six)	5 (17, 29, 41, 53, 65, 77)
20 Node Brick u-p Element (Order One, Two, Three, Four, Five, Six)	6 (18, 30, 42, 54, 66, 78)
20 Node Brick u-p-U Element (Order One, Two, Three, Four, Five, Six)	7 (19, 31, 43, 55, 67, 79)
27 Node Brick Element (Order One, Two, Three, Four, Five, Six)	8 (20, 32, 44, 56, 68, 80)
27 Node Brick u-p Element (Order One, Two, Three, Four, Five, Six)	9 (21, 33, 45, 57, 69, 81)
27 Node Brick u-p-U Element (Order One, Two, Three, Four, Five, Six)	10 (22, 34, 46, 58, 70, 82)
Variable Node Brick Element (Order One, Two, Three, Four, Five, Six)	11 (23, 35, 47, 59, 71, 83)
Variable Node Brick u-p Element (Order One, Two, Three, Four, Five, Six)	12 (24, 36, 48, 60, 72, 84)
Variable Node Brick u-p-U Element (Order One, Two, Three, Four, Five, Six)	13 (25, 37, 49, 61, 73, 85)
8 Node Cosserat Brick Element	96
Bonded Contact/Interface/Joint Element	102
Force Based Dry Hard Contact/Interface/Joint Element	86
Force Based Dry Soft Contact/Interface/Joint Element	87
Force Based Coupled Hard Contact/Interface/Joint Element	97
Force Based Coupled Soft Contact/Interface/Joint Element	98
Stress Based Dry Hard Contact/Interface/Joint Element with Elastic Perfectly Plastic Shear Behavior	99
Stress Based Dry Hard Contact/Interface/Joint Element with Nonlinear Hardening Shear Behavior	100
Stress Based Dry Hard Contact/Interface/Joint Element with Nonlinear Hardening and Softening Shear Behavior	101
Stress Based Dry Soft Contact/Interface/Joint Element with Elastic Perfectly Plastic Shear Behavior	107
Stress Based Dry Soft Contact/Interface/Joint Element with Nonlinear Hardening Shear Behavior	108
Stress Based Dry Soft Contact/Interface/Joint Element with Nonlinear Hardening and Softening Shear Behavior	109
Stress Based Coupled Hard Contact/Interface/Joint Element with Elastic Perfectly Plastic Shear Behavior	104
Stress Based Coupled Hard Contact/Interface/Joint Element with Nonlinear Hardening Shear Behavior	105
Stress Based Coupled Hard Contact/Interface/Joint Element with Nonlinear Hardening and Softening Shear Behavior	106
Stress Based Coupled Soft Contact/Interface/Joint Element with Elastic Perfectly Plastic Shear Behavior	110
Stress Based Coupled Soft Contact/Interface/Joint Element with Nonlinear Hardening Shear Behavior	111
Stress Based Coupled Soft Contact/Interface/Joint Element with Nonlinear Hardening and Softening Shear Behavior	112



(a) Full Model



(b) Only Fiber Beam-Column Elements

Figure 1.33: An example of using threshold to visualize fiber beam-column elements in a soil-structure model.

Bibliography

U. Ayachit. *The ParaView Guide: A Parallel Visualization Application*. Kitware, Inc., USA, 2015. ISBN 1930934300, 9781930934306.

T. H. Group. HDF5. https://www.hdfgroup.org/HDF5/, 2020.

B. Jeremić, Z. Yang, Z. Cheng, G. Jie, N. Tafazzoli, M. Preisig, P. Tasiopoulou, F. Pisanò, J. Abell, K. Watanabe, Y. Feng, S. K. Sinha, F. Behbehani, H. Yang, H. Wang, and K. D. Staszewska. *Nonlinear Finite Elements: Modeling and Simulation of Earthquakes, Soils, Structures and their Interaction*. Self-Published-Online, University of California, Davis, CA, USA, 1989-2025. ISBN 978-0-692-19875-9. URL: http://sokocalo.engr.ucdavis.edu/~jeremic/LectureNotes/.